

Saimaa University of Applied Sciences
Faculty of Technology Lappeenranta
Bachelor of Engineering
Mechanical Engineering and Production Technology

Benjamin Grunwald

Vibration Analysis of Shaft in SolidWorks and ANSYS

Thesis 2018

Abstract

Benjamin Grunwald

Vibration Analysis of Shaft in SolidWorks and ANSYS, 53 pages, 2 appendices

Saimaa University of Applied Sciences

Faculty of Technology Lappeenranta

Bachelor of Engineering

Mechanical Engineering and Production Technology

Thesis 2018

Instructor: Lecturer Seppo Toivanen, Saimaa University of Applied Sciences

The objective of this thesis was to explore the differences in use of a professional FEM-Software and a FEM-Simulation-Tool that is integrated in a 3D-Design software. The study was commissioned by Seppo Toivanen of Saimaa University of Applied Sciences.

In order to determine the differences a vibration analysis of an output shaft of an electric motor with an attached planetary gear was carried out. Three different cases were simulated in each software. For every simulation case a control calculations was carried out to verify the results of the simulation. The information for this thesis was collected from books, internet sources, journals and the Help Guides of each software.

The results of this study show that both software can deliver adequate results but the professional FEM-Software offers more possibilities to reproduce the reality within the software. Therefore it is recommended to use the professional FEM-Software for special and complex cases. Simulation of those cases requires highly skilled engineers, who know exactly what they do. In return the simulation tool is excellent for beginners in FEM to learn the basics and for quick analysis in between the designing work. For further investigations it is recommended to spread the study over several analysis types and to analyse several structures with different boundary conditions.

Keywords: Finite Elements Method - FEM, Finite Elements Analysis - FEA, vibration analysis

Table of contents

1	Introduction	5
1.1	Background.....	5
1.2	Objectives	6
1.3	Thesis format	6
2	Theory review	8
2.1	General procedure in FEM simulation.....	8
2.1.1	Engineer's procedure	8
2.1.2	Software procedure	9
2.2	Elements.....	11
2.3	Mass models.....	12
2.4	Solvers.....	13
2.4.1	Direct solvers	13
2.4.2	Iterative solvers	13
2.4.3	Solvers for vibration problems	14
2.4.4	The choice of the solver	14
2.5	Static linear analysis	16
2.6	Vibration analysis.....	17
2.6.1	Eigenfrequency	18
2.6.2	Eigenmode	19
2.7	Buckling analysis	20
3	Simulation	23
3.1	Preparation of the models.....	23
3.1.1	Preparation of the shaft	23
3.1.2	Preparation of the shaft with reduced mass moment of inertia.....	24
3.2	Simulation in ANSYS	26
3.2.1	Simulation of the shaft.....	27
3.2.2	Simulation of the shaft in the first gear	33
3.2.3	Simulation of the shaft in the second gear	36
3.3	Simulation in SolidWorks	37
3.3.1	Simulation of the shaft.....	37
3.3.2	Simulation of the shaft in the first gear	39
3.3.3	Simulation of the shaft in the second gear	39
4	Comparison of ANSYS and SolidWorks	41
4.1	Material library	41
4.2	Mesh and element type.....	41
4.3	Boundary conditions	42
4.4	Solvers.....	43
4.5	Results.....	43
5	Conclusion	45
5.1	Limitations.....	45
5.2	Recommendations.....	45
5.3	Reflections	45
5.4	Final conclusion	46
	References.....	48
	Appendices	51

1 Introduction

This thesis investigates the differences of a professional FEM-Simulation-Software and a FEM-Simulation-Tool integrated in a 3D-Design-Software regarding their easiness of use. In order to determine the differences, a vibration analyses of an output shaft and the attached planetary gear of an electric motor is carried out. ANSYS Workbench 18.2 is used as the professional FEM-Simulation-Software, while the FEM-Simulation-Tool is used from SolidWorks 2017/2018. Both software editions are the latest versions available at the time the thesis was started.

1.1 Background

The Finite Elements Method or short FEM is a comfortable way for engineers to carry out complex calculations in mechanics and other physical fields. It enables the engineers to make accurate estimations of stress, strain, frequencies and much more. Not only analyses of single parts are possible, but also of subassemblies and complete assemblies. It is even used to simulate crash tests of cars. The Finite Elements Analysis (FEA) is nowadays an invaluable tool in the product development. It is already applied in the state of concept generation and extensively used for the further development of parts. This does not save only a lot of time, but also a lot of money, since the budget for tests and prototypes can be reduced to a minimum. The Finite Element Method is in cooperation with CAD (Computer Aided Design) the most powerful procedure to rationalize and optimize the work of an engineer qualitatively. But it is not advisable to have a blind trust into FEM. The basics of mechanics and the basics of the FEM- procedure must be understood by the calculation engineer. Otherwise a calculated structure could end up in a disaster in real life, which often results in deaths. The calculation engineer is liable and has to be aware of his responsibility.

There is an abundance of FEM-Simulation-Software available on the market. Common Software are ANSYS, ABAQUS, NASTRAN or I-DEAS. These Software are capable of simulating with practically no limitations, with regard to the size of the structure as well as the reproduction of reality. But there is not only pure FEM-Simulation-Software. FEA is also possible in 3D-CAD-Software and it

plays a more and more important role. Therefore, the integrated simulation tools are permanently improved and the amount of simulation options increases steadily.

1.2 Objectives

The objectives of this thesis are simple. First: Finite Elements-Analyses are to be carried out in a proper way to get results and detect differences within both software. Secondly, the results are to check for plausibility by means of a control calculation and the behaviour of the structures mode shape. The third objective is to compare the two software regarding the detected differences that were encountered during the simulation work.

1.3 Thesis format

In order to carry out this thesis, a lot of preparatory work was done. Specifically the preparation of the author's own knowledge in the field of vibrations and FEM theory was required. Additionally it was necessary to do some analysis exercises to get along with both software, particularly with ANSYS. A good understanding of basic mechanics and machine dynamics is essential to interpret the simulation result and to carry out a control calculation. But even more important is the understanding of the principle operation of Finite Elements Method and the different ways to solve the matrices. This thesis contains necessary knowledge in the following fields:

- Principle procedure in FEA
- Calculation models (Elements)
- Solving algorithms
- Basic mechanics with regard to FEA

The thesis is broken down into three main tasks: Theory review, Simulation and Comparison of ANSYS and SolidWorks.

Task 1: Theory Review

The theory part shows how a structure is prepared by the software to generate the matrices and points out different ways to solve them. A very deep mathematical point of view with long derivations and differential equations is omitted. Additionally some basics of mechanics regarding FEM are looked at.

Task 2: Simulation

This part involves the preparation of the model before the simulation and the selection of the settings within both software. It also contains verification of the results by means of control calculations and checking the behaviour of the model in the simulation.

Task 3: Comparison of ANSYS and SolidWorks

The last section points out the differences between the professional simulation software and the simulation tool that were detected during the simulation work.

2 Theory review

2.1 General procedure in FEM simulation

Not only the FEM-Software uses a procedure to carry out an analysis. It is also advisable for the calculation engineer to follow a certain procedure in order to avoid mistakes, since the FEA is not as easy as it seems.

2.1.1 Engineer's procedure

Before a FEM-Simulation can be carried out a CAD-Model has to be created. Depending on the solver and its compatibility with CAD-Data types, it may be that the CAD-Model has to be interfaced with the solver. This works with standard interfaces like IGES (Initial Graphics Exchange Specification) or STEP (Standard for the Exchange of Product Model Data). It is also possible to generate the geometry in the simulation software. However, it is advisable to take the original CAD-Data for the simulation in order to reduce the working hours. The following procedure is generally the same but may vary a bit in different analysis types. (Klein, 2014)

1. Edit the 3D Geometry

It is necessary to reduce the geometry details like small holes, radii or ledges to a minimum in order to reduce the faults in the calculation. Because these small features increase the amount of bad elements and distort the results.

2. Choose the right material

In every simulation software and every simulation tool there is a wide range of materials. But sometimes it happens that the library does not contain the required material. Then a material with similar properties must be searched for. The most attention has to be paid to the properties that play a specific role for the particular analysis that is to be carried out.

3. Choose the element type

Choosing an appropriate element type for the numeric calculation is one of the most essential steps in the process-chain of the simulation. Depending on the type of analysis, it is necessary to choose the element type with regard to the desired results and their location on the component.

4. Create a Mesh

Before creating the mesh it has to be considered if there is any possibility to use a symmetry in the component. Then it is possible to save time by creating a proper mesh only for a half or a quarter of a section and mirror it. The pre-processor usually creates an appropriate mesh automatically, but sometimes the mesh needs to be refined at some certain spots. A local refinement in the mesh should be considered where more accurate results are required, for instance in the area of some holes.

5. Constraints

Setting the right constraints is the most important part in this procedure and has the highest influence to the results. Therefore it is crucial to think about the real behaviour and bring the constraints in the simulation close to reality.

6. Forces

The elements are connected over their nodes therefore the forces should be applied to the nodes, if possible and if forces are needed for the simulation.

7. Solving

If all the previous steps have been carried out the simulation can be run and the solver can compute the results.

8. Checking the results and deformation behaviour

After the results were calculated, it is important to check them by means of an analytical control calculation. It has to be checked whether the simulated value is close to the analytically calculated value. In analyses of vibrations the analytical results can be used as an upper or lower limit to check the value. It is also important to check the deformation behaviour in order to figure out if there are any mistakes in the constraints or loads.

2.1.2 Software procedure

The following steps describe the general procedure of the simulation software in a linear static analysis. This is supposed to give an idea of how the software proceeds a simulation in general. The procedure is shown exemplary on a truss element, but works similar with all other elements. (Dominico, 2017)

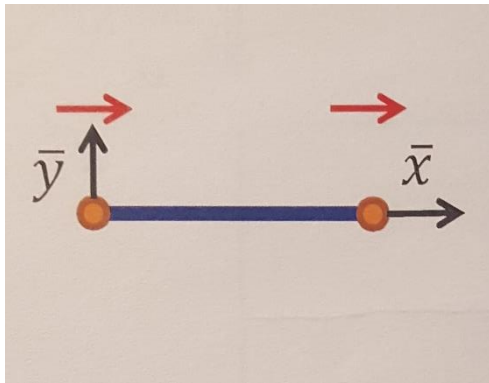


Figure 1 Local coordinate system (Dominico, 2017)

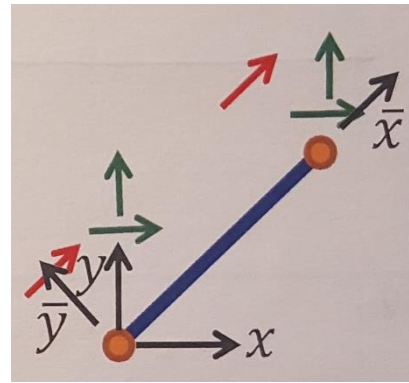


Figure 2 Global coordinate system (Dominico, 2017)

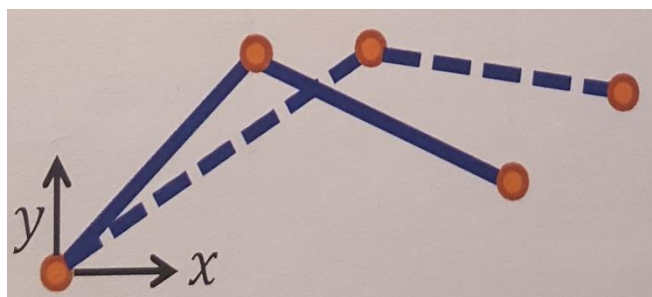


Figure 3 Assembled structure (Dominico, 2017)

- 1 Calculating the stiffness matrix for each element in a local coordinate system (\bar{x}, \bar{y}) : $\rightarrow K_{element,local}$ see Figure 1
- 2 Transformation into a global coordinate system (x, y) : $\rightarrow K_{element,global}$ see Figure 2
- 3 Assembling all elements according to the meshed structure $\rightarrow K_{structure}$ see Figure 3
- 4 Add loads and constraints to the structure $\rightarrow K_{structure} * u = f$
- 5 Solving the system of equations for forces, displacements, strain and stress
- 6 Post-processing (colouring the elements according to the value of stress and strain)

2.2 Elements

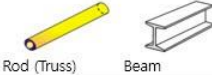
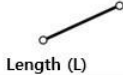



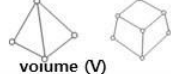
Type	Actual Models	Finite Element Expressions (Geometric Properties Defined by Nodes)	Additional Requirements (Actual Volume Calculation)
1D	 Rod (Truss) Beam	 Length (L)	Area (A, cross-sectional shape) $\rightarrow V = L \times A$
2D	 Shell, Plane Stress, Plane Strain, Axisymmetric, etc	 Area (A)	Thickness (t) $\rightarrow V = A \times t$
3D	 Solid	 volume (V)	None (volume calculation possible)

Figure 4 Element types (Rusu, 2013)

1D elements are distinguished into two types, truss elements and beam elements. Whereas truss elements can only forward forces along their longitudinal axis, beam elements can also forward transverse forces and moments. Hence truss elements have only one translational DOF (degree of freedom) at each node and beam elements have two translational and one rotational. 1D elements are used for instance simulate frame structures or shafts. (Dominico, 2017)

2D shell elements are also distinguished into two types, Tri-Elements and Quad-Elements. Quad-Elements should be preferably used, since their stress calculations are done by a linear function. Then another distinction is made between shell elements that are only used for stress simulation and for stress and bending simulation. They differ in their number of DOFs. Shell elements are used to simulate sheet metal structures. (Dominico, 2017)

3D solid elements or also often called continuum elements have only three translational DOFs at each node, no rotational degrees of freedom. Also here applies the rule that elements with a quad shape should be preferably used. The solid types are used to simulate thick walled structures like an engine. (Dominico, 2017)

The previous described elements are automatically fitted together to one big so-called mesh, what extends to the entire part. But the automatically generated mesh is not always perfect. In some analysis types, like linear static, local refinements should be considered for the areas that are of special interest. (Dominico, 2017)

2.3 Mass models

In dynamic FEM-Simulations there are two basic mass models that can be applied to assemble the mass matrix for the elements. The first and most common used mass model is the so-called lumped mass. In the lumped mass approach the mass of every element is equally distributed to its nodes. In the case of a beam element, half the mass of the element is assumed to be concentrated at each node. The connecting material is treated as a massless spring with axial stiffness. For a continuum element the mass is determined via volume of the element and the material density. The procedure of the mass distribution is the same. The following figure demonstrates the mass distribution in a structure that is meshed with tetrahedrons. When all masses are distributed to the nodes, the mass matrix is assembled. In a lumped mass matrix, there are only numerical values on the main diagonal. This offers computational advantages for numerical operations. (Hutton, p. 402- 407, 2004; Rieg, et al., p.150- 151, 2014; CAEFEM GmbH, 2015)

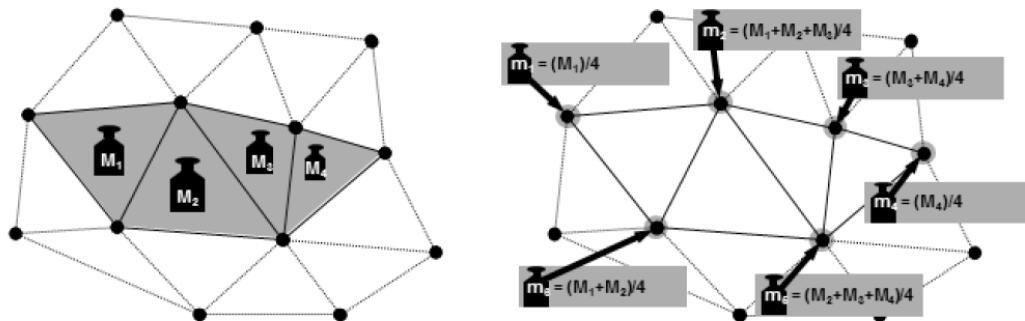


Figure 5 Mass distribution tetrahedron elements (Rieg, et al., p.150- 151, 2014)

The other mass model is the so-called consistent mass, where the influence of the distributed mass is considered in the element. The mass matrix is assembled in a similar way to the stiffness matrix. In comparison to the lumped mass matrix, the consistent mass matrix is computationally more difficult to handle, since there are not only numerical values on the main diagonal. The consistent mass model can be used to obtain upper bounds for the natural circular frequencies. (Hutton, p. 402- 407, 2004; CAEFEM GmbH, 2015)

2.4 Solvers

In the Finite Elements Method the solvers have a specific importance. For instance if there is a structure with 1000 degrees of freedom, it means that a matrix with 1.000×1.000 variables has to be solved. This is a rather small problem nowadays. If the structure gets bigger in a three dimensional problem, the number of degrees of freedom increases exponentially. This makes clear that an analytical solution is inefficient and nearly impossible for FE-Analyses. Due to the size of the systems of equations, numerical methods are used in order to solve the equations. A distinction is made between direct and iterative methods and in dynamics also reduction methods. It is renounced to show the solvers from a very deep mathematical point of view. Only the advantages and disadvantages are supposed to be shown out in this chapter. (Klein, 2014; Rieg, et al., 2014)

2.4.1 Direct solvers

All direct solvers are based on the so-called Gauss methods.

The Gaussian-Elimination-Method works in the way that fundamental transformations change the matrix but the solution is kept. This method is a very fast direct solver. It is only a stable process for positive definite matrices without an additional job step. That means the accuracy for not positive definite matrices can get rather poor. (Wikimedia Foundation Inc., 2007)

The Cholesky-Decomposition is related to the Gaussian-Elimination but much more numerically stable. It is also carried out much faster because it needs less calculating operations. A big disadvantage is that it works only for positive definite matrices. (Rieg, et al., 2014)

2.4.2 Iterative solvers

This kind of solvers carry out the same calculation again and again, until the solver has reached the stop criterion. Iterative solvers work much faster than the direct solvers. This is not due to their mathematical structure, but to the combination with the non-zero elements storage process. That means that less mathematical operations like summation and multiplication have to be carried out. Additionally less storage space on the computer is needed. Another advantage is

that they are almost immune to inappropriate numbering of nodes. But a major disadvantage is that they need a stop criterion in order to stop the iteration process. It is not really clear when the solver has reached an “exact” solution. It might happen that the iteration process is stopped too early and the result is not yet accurate enough. That might lead to a misinterpretation in the results. (Rieg, et al., 2014; Wikimedia Foundation Inc., 2018)

The basic CG Solver or conjugated gradient solver is a very useful method for Finite Element iteration solvers. It delivers the solution of a system of equations in n unknowns after at most n steps, with an “exact” solution. That means if the structure has 100.000 degrees of freedom, a reasonable solution is achieved after approximately 100.000 iteration steps. This proceeds fairly fast and needs only a very little storage space. But there is a way to proceed faster. (Rieg, et al., 2014)

The Preconditioned CG Solver is much faster than the Basic CG Solver and hence the fastest one among all solvers. In return it needs a little more storage space than the basic CG. With this solver one system of equations is solved in every iteration step. Due to the preconditioning, the system of equations is solved as efficiently and simply as possible. There are also different ways of preconditioning, but they are not further described in this point. (Rieg, et al., 2014)

2.4.3 Solvers for vibration problems

Due to the desired eigenvalue the solvers for vibration problems differ a bit from the other ones. The most commonly used solvers are Jacobi Method, Givens Method and Householders Method for very small problems, what means <2500 degrees of freedom. For bigger problems the most common ones are subspace iteration, simultaneous iteration, Lanczos’ Method, which reduce the number of the degrees of freedom by transformations. (Petyt, 2010)

2.4.4 The choice of the solver

The solver is mostly selected automatically by the software, but in some cases it makes sense to select the solver manually. The choice of the solver is then basically guided by the total number of DOFs in order to have an efficient calculation

process. The most efficient direct methods are the Gaussian-Elimination- and the Cholesky-Decomposition-Method. These methods are only used for smaller problems, because they need a lot of storage space. For big problems with more than 50.000 unknown variables, iteration methods like the Conjugate Gradient (CG) Method and the Preconditioned Conjugated Gradient Method (PCG) have come out on top. They need less storage space and compute much faster than the direct methods. (Klein, 2014)

For a 3 dimensional problem the generated mesh has n nodes. In a pure mechanical problem the three unknown for each node are solved. That means the displacements along x-, y- and z-directions. Hence, the total number of degrees of freedom can be calculated by the following equation (Dölle & Heiny , n. d.):

p = number of unknowns for each node

n = number of nodes

$$n_{dof} = p * n \quad (1)$$

The following table gives an overview about a few common solvers, their required storage space and the needed calculating time.

Unknown variables N		Very small problems N=1.000	Small problems N=3.600	Medium-sized problems N=14.000	Big problems N=56.000	Very big problems N=220.000
Method						
Cholesky	T	0,001 s	0,11 s	1,70 s	22 s	-
	Ss	185 kB	1,31 MB	10 MB	79 MB	-
Gauss-Seidel	T	170 s	3.000 s	-	-	-
	Ss	49 kB	193 kB	-	-	-
CG	T	0,36 s	4,5 s	92 s	950 s	8.000 s
	Ss	67 kB	250 kB	985 kB	3,8 MB	15 MB
PCG	T	0,05 s	0,1 s	1,95 s	3,5 s	17,5 s
	Ss	70 kB	250 kB	1,2 MB	4,6 MB	20 MB
Fontal solver	T	1.000 s	3.600 s	-	-	-
	Ss	50 kB	200 kB	-	-	-

Table 1: Efficiency of solving algorithms (T= needed computation time, Ss= needed storage space) (Klein, 2014)

The total numbers of degrees of freedom is the crucial value for solvers for vibration analyses as well. See chapter 2.4.3.

2.5 Static linear analysis

A static linear analysis is used to determine forces and stresses within structures. A linear problem requires that only small deflections appear at the structure, that there is no contact between parts and only elastic material behaviour is considered. The basic equation of the static linear problems is (Dominico, 2017):

k = stiffness

$[K]$ =stiffness matrix

u = displacement

U = displacement vector

f = force

F = force vector

$$k * u = f \quad (2)$$

Or written as matrix equation:

$$[K] * U = F \quad (3)$$

For each degree of freedom either u or f is known but never both values. If a degree of freedom is fixed ($u=0$), then f is unknown and has to be calculated. But if the degree of freedom is not fixed, the force can be defined and the displacement has to be calculated. The stresses can be finally calculated with the displacements. (Dominico, 2017)

The stiffness matrix defines the stiffness for the entire structure that has to be examined. It has as many rows and columns as the whole structure has degrees of freedom. The derivation of such a matrix is rather complex and very mathematical, therefore it is omitted at this point. The following matrix on the left-hand side describes a stiffness matrix for the simplest structure that can be used in FE-Analysis, the truss element. It has two degrees of freedom. The matrix on the right-hand side describes the stiffness matrix of one simple beam element with six degrees of freedom. This simple example shows that the matrices for a whole structure get very big and complex. (Dominico, 2017)

$$K_T = \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad K_B = \frac{E}{l} \begin{bmatrix} A & 0 & 0 & -A & 0 & 0 \\ 0 & \frac{12I}{l^2} & \frac{6I}{l} & 0 & -\frac{12I}{l^2} & \frac{6I}{l} \\ 0 & \frac{6I}{l} & 4I & 0 & -\frac{6I}{l} & 2I \\ -A & 0 & 0 & A & 0 & 0 \\ 0 & -\frac{12I}{l^2} & -\frac{6I}{l} & 0 & \frac{12I}{l^2} & -\frac{6I}{l} \\ 0 & \frac{6I}{l} & 2I & 0 & -\frac{6I}{l} & 4I \end{bmatrix}$$

There are two requirements to the matrix that have to be fulfilled for a static linear analysis. The first one is that all elements on the diagonal of the stiffness matrix must have a positive sign. Because the stiffness combines load and deformation. A load at a DOF leads to a deformation in the direction of the load. If there was a negative sign, the deformation would be in the opposite direction of the force direction. Secondly, a stiffness matrix must be symmetrical regarding its main diagonal. This describes the symmetrical behaviour and is true for all linear systems. (Dominico, 2017)

2.6 Vibration analysis

Vibrations are widespread in technical applications, especially in structures that are in motion or undergo dynamic loads. In mechanical vibrations there are many types of vibrations and a lot of values to characterize them. Two that are of big interest for engineers and often simulated with the Finite Elements Method are the eigenfrequency and the eigenmode.

There are different kinds of vibrations in machine dynamics. The most common vibrations that a shaft can undergo are bending vibrations and torsional vibrations. Bending vibrations occur in a shaft because of unbalanced mass. Shafts do not have a perfect round shape and the revolving axis does not match exactly the centre axis of the shaft. So when the shaft revolves around its axis and speeds up, it reaches a point where vibrations arise. This point is the circular eigenfrequency. If the speedup continues the vibrations disappear. Torsional vibrations can occur if a mass like a pulley or a gear wheel is attached to the shaft. Then a torsion takes place along the centre axis. (Holzweißig & Dresig, 1982)

2.6.1 Eigenfrequency

The so-called natural frequency or eigenfrequency is a very important value of structures that carry dynamic loads. Due to the physical phenomenon of resonance it happens that a structure undergoes very heavy vibrations if it faces frequencies with the same value as its eigenfrequency. These frequencies lead mostly to a catastrophic failure in the structure. Therefore analyses of eigenvalue problems are significantly important in mechanical engineering. (Liu & Queck, 2003)

The following equations are supposed to show how this phenomenon appears:

k = stiffness

m = mass

ω = frequency in rad/s

$[K]$ = stiffness matrix

$[M]$ = mass matrix

The basic equation for the eigenfrequency is (Liu & Queck, 2003):

$$\omega = \sqrt{\frac{k}{m}} \quad (4)$$

(4) transformed to k :

$$k = \omega^2 * m \quad (5)$$

(5) transformed to 0, written as a matrices equation and multiplied by the amplitude ϕ (Liu & Queck, 2003):

$$[[K] - \omega^2 * [M]]\phi = 0 \quad (6)$$

Equation (6) is the so-called eigenvalue equation. The variable ω^2 can be replaced by λ . For certain values of λ the determinate of the equation (7) disappears.

$$|[K] - \lambda * [M]| = 0 \quad (7)$$

This means that the stiffness of the initial rigid body disappears and it becomes flexible like rubber. The value of λ for which this equation becomes true is the eigenfrequency. (Liu & Queck, 2003)

2.6.2 Eigenmode

By substitution of ω by λ_i ($i=1, 2, 3, 4, \dots$) and the vector ϕ by ϕ_i in the eigenvalue equation (6) the following equation is obtained (Liu & Queck, 2003):

$$|[K] - \lambda_i * [M]|\phi_i = 0 \quad (8)$$

The vector ϕ_i is the so-called eigenvector and corresponds to the i th eigenfrequency. This eigenvector belongs to a certain vibration mode that determines the vibrating shape of a structure. Thus the i th eigenfrequency evokes the i th eigenmode. The shapes of the single modes can be totally different as Figure 6 to 9 show. Eigenmodes are another important characteristic of a structure. Eigenvectors can be used mathematically to construct the displacement fields. (Liu & Queck, 2003)

The following pictures show the vibration behaviour of a metal construction and a connecting rod in certain modes to point the mostly unpredictable shapes up:

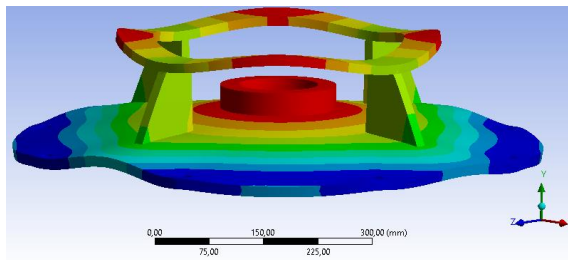


Figure 6 1st mode of a plate construction

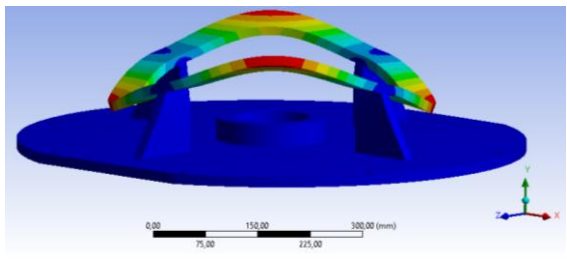


Figure 7 2nd mode of a plate construction

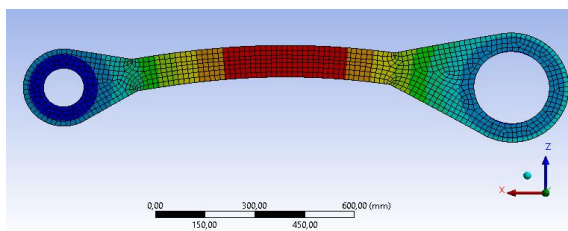


Figure 9 1st mode of a connecting rod, front view

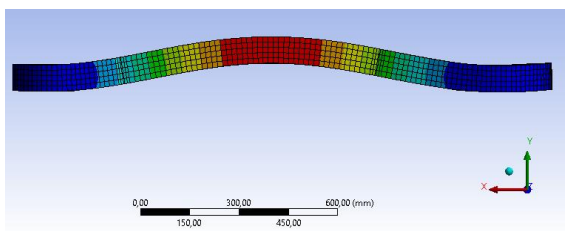


Figure 8 2nd mode of a connecting rod, top view

2.7 Buckling analysis

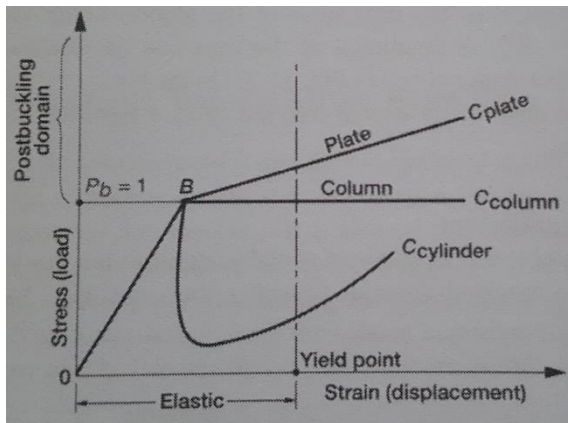


Figure 10 Buckling behaviour of structures (Pataik & Hopkins, 2004)

Buckling occurs when a structure suffers a compressive stress. The structure can withstand a certain compressive stress depending on its cross section and on the constraints of the technical problem. If the load on a structure increases the stress in the cross section increases linear up to a certain point (point B in Figure 10), the so-called bifurcation point. This point also

carries the name buckling point or critical point, because at this point the structure starts buckling. The behaviour of the structure beyond this point differs due to its shape of the cross section. Some examples of the behaviours (the line between B and C) are shown in the stress-strain-diagram in Figure 10. (Pataik & Hopkins, 2004)

Buckling is also referred to as a stability problem and can be classified into three categories, a stable equilibrium, an unstable equilibrium and a neutral equilibrium. The line in Figure 11 represents a ground contour with a valley at point A, a hill at point B and a flat area at point C. The first equilibrium is the stable equilibrium. If a small perturbation is applied to the ball at point A, it will return to its initial position after a few oscillations.

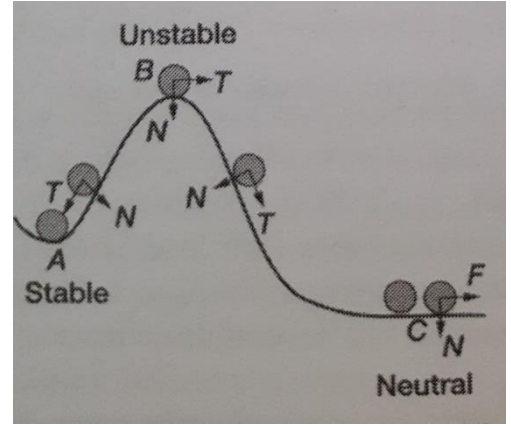


Figure 11 State of stability (Pataik & Hopkins, 2004)

Practically a plate-buckling problem can be considered for this category. The second one, the unstable equilibrium can be found at point B. Here, only a small perturbation is required to move the ball and destroy the equilibrium. A practical case for this class could be a cylinder-buckling problem. The last one is the neutral equilibrium at point C. When the ball at this point is slightly perturbed, it rolls gently over the flat ground and stops. In this case the ball stays at the perturbed location, therefore it is called the neutral equilibrium. A column-buckling problem is a practical example of this case. (Pataik & Hopkins, 2004)

In the Finite Element Method a static linear analysis is the basis for a general buckling problem. The basic equation for this problem is (9), in which f is an arbitrary load, u the displacement and k the stiffness. Equation (10) is similar to (9) but written as a matrix equation (Steinke, 2015)

$$k * u = f \quad (9)$$

$$[K] * U = F \quad (10)$$

The stresses can be calculated for the applied force, when the displacements are known. The stiffness matrix $[K]$ consists of two parts, the elastic stiffness matrix $[K_E]$ and the so-called geometrical stiffness matrix $[K_G]$ as the next equation shows (Steinke, 2015):

$$[K] = [K_E] + [K_G] \quad (11)$$

The geometric stiffness matrix receives a thrust load as the factor P and the thrust load gets multiplied by the factor λ , what leads to equation (12). The factor λ is

the eigenvalue. Under certain circumstances the determinant of $[K]$ disappears what means that also the stiffness disappears and the structure starts buckling. (Steinke, 2015)

$$|[K]| = \left| [K_E] + \lambda \frac{P}{l} [K_G] \right| = 0 \quad (12)$$

In the solving process, the solver looks for the values that make the determinant disappear.

3 Simulation

3.1 Preparation of the models

The following picture shows the section of the entire electric motor with the light blue coloured shaft, the green coloured rotor of the electric motor with the sun gear and the planetary carrier, which is to simulate. The bearings, which support the shaft, are not included in this model. Some small parts that have no big influence to the final result were neglected in order to simplify the simulation and the belonging control calculations.

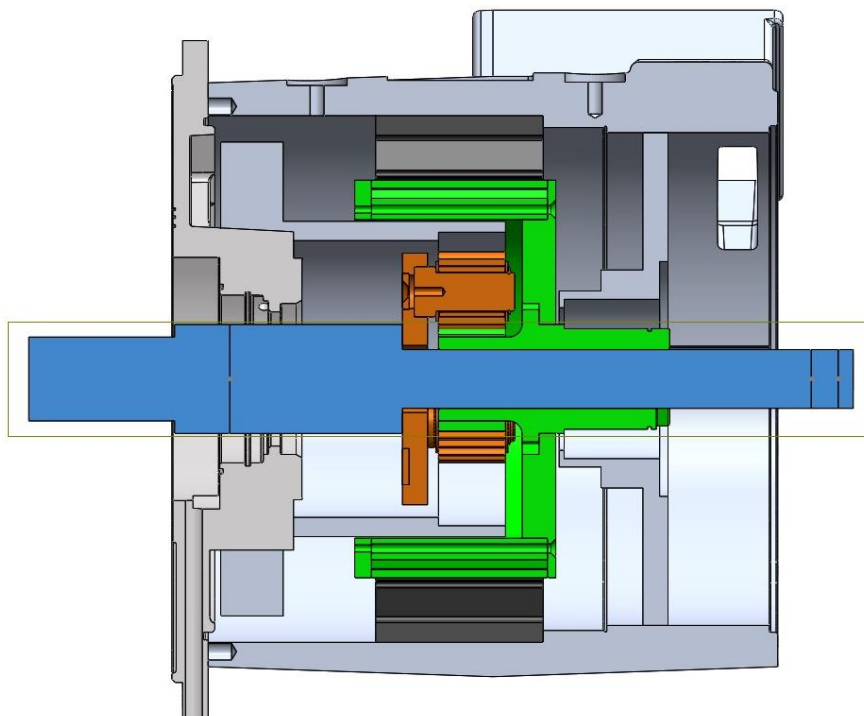


Figure 12 Section view of electric motor

3.1.1 Preparation of the shaft

The following figure shows the single shaft in its full shape. In order to create good elements for an accurate simulation the geometry of the shaft was cleaned. Figure 13 shows the cleaned shaft and its constraints.

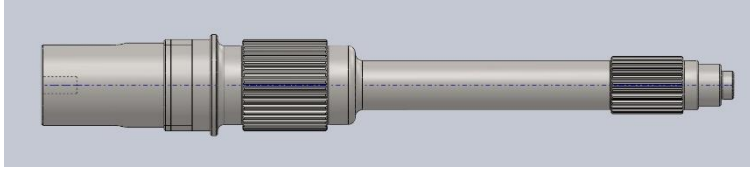


Figure 14 **Uncleaned 3D model of the shaft**

All the small edges, radii and the teeth were deleted to avoid getting elements of bad shape and in order to reduce the calculation time. The cylindrical surfaces on which the bearings are placed were split in the CAD-Software to set the constraints only to the areas that are needed. The bearings are assumed to be infinite stiff. Both bearings are 372 mm far from each other. The left bearing at the bigger diameter is the fixed bearing, the right one at the smaller diameter is the loose bearing as it can be seen in the picture below.

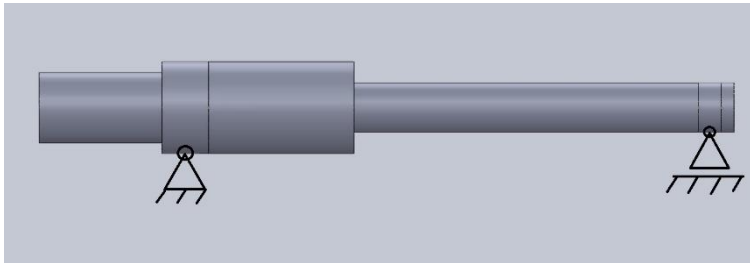


Figure 13 **Cleaned 3D model for simulation**

3.1.2 Preparation of the shaft with reduced mass moment of inertia

The rotor of the electric motor and the planetary gear have to be replaced for the simulation. Therefore their mass moments of inertia are reduced to a single mass moment of inertia. This is necessary since the rotational speed of the rotor, the input shaft and the sun gear (green in the figure below) differs from the rotational speed of the planet carrier and its planet gears (orange in the figure below). The mass and the mass moment of inertia of both assemblies were read from mass properties in the CAD- software. In order to obtain the reduced mass moment of inertia the following procedure was carried out.

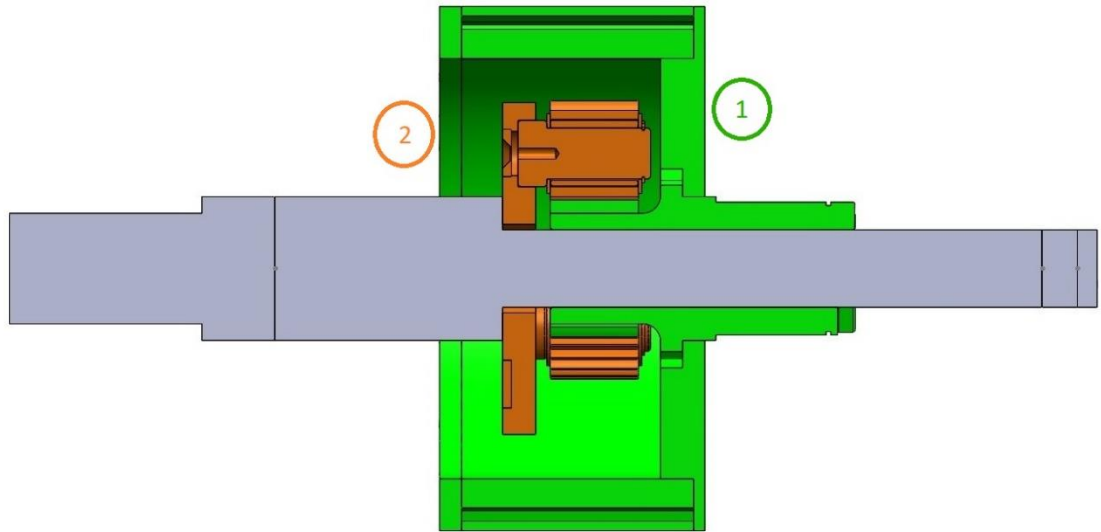


Figure 15 Model for reduction of mass moment of inertia

First, the stationary gear ratio i_0 and the gear ratio i between the fixed sun gear and the planet carrier of the planetary gear have to be calculated by (Weidner, 2016):

$$i_0 = -\frac{teeth_{ring\ gear}}{teeth_{sun\ gear}} \quad (13)$$

$$i = 1 - i_0 \quad (14)$$

$$i = 1 - \left(-\frac{76}{29}\right) = \underline{\underline{3,6207}}$$

The next step is to calculate the reduced mass moment of inertia by (Heid Antriebstechnik Produktion & Handel GmbH, n. d.):

$$J_{red} = J_1 + \frac{J_2}{i^2} \quad (15)$$

$$J_{red} = 140905,28\ kgmm^2 + \frac{7187,46\ kgmm^2}{3,6207^2}$$

$$J_{red} = \underline{\underline{141457,7\ kgmm^2}}$$

The model for the simulation is basically the same as the previous one for the blank shaft. The only difference is that the mass moment of inertia around the Z-axis is considered by means of an inserted point mass. The mass moment of inertia around the X- and Y-axis is not considered, since the rotor is attached to

the input shaft and the input shaft is supported separately. The introduction of the mass moment of inertia into the shaft takes place over the toothing near the bearings. The exact position of the point mass is:

$$l_5 = 87 \text{ mm} - 40 \text{ mm} / 2 - 17 \text{ mm}$$

$$l_5 = \underline{50 \text{ mm}}$$

Whereat 40 mm is the width of the counter part of the jaw clutch and 17 mm is the dimension of the overlaying shaft end on the right side, measured from the middle of the bearing. A clearer overview gives the drawings of the shaft and the counterpart of the jaw clutch in the appendices.

If the electric motor is driven in the second gear, only the rotational speed of the shaft and the position of the introduction of the mass moment of inertia change. The reduced mass moment of inertia does not change, because the input rotational speed remains constant. The position of the point mass is then:

$$l_8 = 289 \text{ mm} - 17 \text{ mm} + 40 \text{ mm} / 2$$

$$l_8 = \underline{292 \text{ mm}}$$

The distance l_8 is the exact distance from the loose bearing to the point mass. The 17 mm describe again the dimension of the overlaying shaft on the right side, from the middle of the bearing. The dimension 40 mm is the width of the counterpart of the jaw clutch and 289 mm is the distance from the right end of the shaft to the beginning of the toothing.

3.2 Simulation in ANSYS

For the simulation in the ANSYS software, the models were created in SolidWorks and according to 3.1.1 edited. Finally they were saved as an IGES- file. When they were imported into the ANSYS software the IGES-geometries had to be edited in ANSYS SpaceClaim. This was necessary to use proper constraints.

3.2.1 Simulation of the shaft

The first step in the simulation is always to edit the geometry in order to get a small amount of bad elements. This step was already described in chapter 3.1.1.

The actual material of the shaft is 42CrMo4, but the ANSYS material library does not offer such a wide range of materials. Therefore, the material for the simulation was selected as a structural steel because it matches the mechanical properties the best and is sufficiently accurate.

The element type Hex dominant is chosen for the meshing method. That means for this method the Hex elements are preferably used, but also other elements in order to fill the missing geometry as good as possible. The initial element size is set to 12mm per element, other settings are not made. After solving the model the size of the elements will be adjusted to check the sufficiency of the elements. The number of elements is now at 1876.

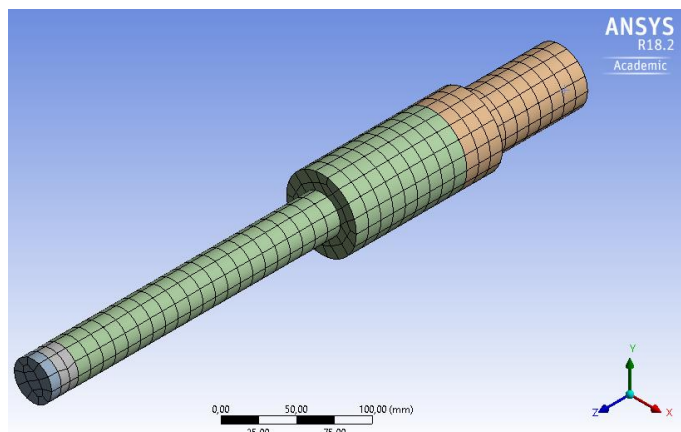


Figure 16 **Meshed shaft**

For the both bearings the boundary condition Cylindrical Support is used, since this kind of boundary condition is well suitable for a simulation of round parts that move relatively to its environmental parts. The motion can be translational as well as rotational to the surrounding parts. This boundary condition prevents cylindrical surfaces from moving and deforming in the constraint directions. For the fixed bearing the axial and the radial DOF is constraint, the tangential DOF is free. For the loose bearing, only the radial DOF is constraint, the axial and tangential DOFs are free. The entire shaft is now free to rotate. (ANSYS Inc., 2017)

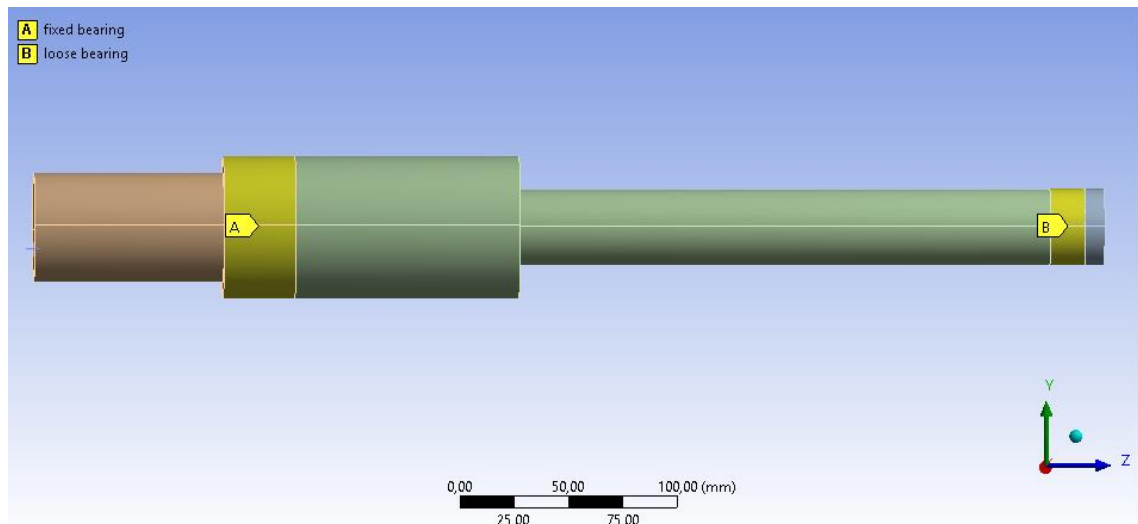


Figure 17 Supports of the shaft

Forces, moments or other boundary conditions do not play any role for this simulation of natural frequencies.

The solver was run and the simulation shows for the first mode the frequency 0 Hz. This is the so-called rigid body mode and occurs for every degree of freedom of the system that is not constraint. In this case it is the revolution around the centre axis of the shaft. The second mode shows the value of 1635,8 Hz, what is the first natural frequency. The mode shape shows the expected bent shape between the two bearings. In the animated mode shape, the elongated rod moves up and down. A closer look to the bearings shows, that the elements in the loose bearing move slightly along the centre axis of the shaft. When the shaft bends downwards, the upper elements in the bearing move to the left, whereas the bottom elements move to the right. In return, the elements in the fixed bearing remain in their initial position. This shows that both boundary conditions behave as they should. The third mode shows a frequency of 1638,3 Hz, what is only a difference of 2,5 Hz. The mode shape and the behaviour of the elements in the bearings is the same. The only difference is the direction of the elongation in the mode shape animation. It elongates perpendicular to the first mode. The third mode is basically the same mode as the second one. The fourth mode is a torsional vibration of the thinner diameter at a frequency of 2965 Hz. (ANSYS Inc., 2017)

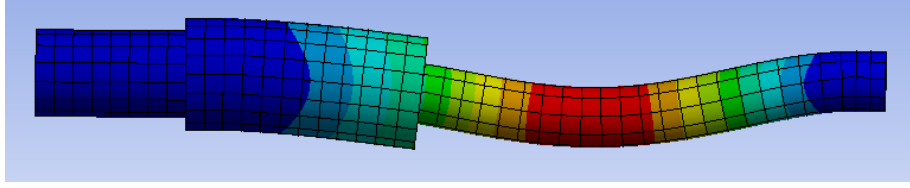


Figure 18 Mode shape of second mode

To check the plausibility of the frequency an approximation calculation for the critical speed according to Dunkerley is carried out.

The approximation formula of Dunkerley is (Haberhauer & Bodenstein, 2009):

$$\frac{1}{\omega_c^2} = \frac{1}{\omega_{kl}^2} + \frac{1}{\omega_{c1}^2} + \frac{1}{\omega_{c2}^2} + \frac{1}{\omega_{c3}^2} + \dots \quad (16)$$

with

$$\frac{1}{\omega_{ci}^2} = \alpha_{ii} * m_i \quad (17)$$

and

$$\frac{1}{\omega_{kl}^2} = \frac{l^4}{d^2} * \frac{2\rho}{\pi E} \quad (18)$$

The influencing factors for masses between the bearings are calculated by (20) and (21) the factor for a mass outside of the bearing is calculated by (19) (Holzweißig & Dresig, 1982):

$$\alpha_{11} = \frac{l_1^2}{c_1(l_2 + l_3 + l_4)} + \frac{(l_1 + l)^2}{c_2(l_2 + l_3 + l_4)} + \frac{l_1^2 l}{3EI_a} + \frac{l_1^2}{3EI_1} \quad (19)$$

$$\alpha_{22} = \frac{(l - l_2)^2}{c_1 l^2} + \frac{l_2^2}{c_2 l^2} + \frac{l_2^2 (l - l_2)^2}{3EI_2 l} \quad (20)$$

$$\alpha_{33} = \frac{l_4^2}{c_1 l^2} + \frac{(l - l_4)^2}{c_2 l^2} + \frac{l_4^2 (l_2 + l_3)^2}{3EI_3 l} \quad (21)$$

All these equations for the influencing factors are tailored for the lengths in Figure 19. c_1 and c_2 are the stiffness of the bearings, but the bearings are considered infinitely stiff, what leads to the following equations:

$$\alpha_{11} = \frac{l_1^2 l}{3EI_a} + \frac{l_1^2}{3EI_1} \quad (22)$$

$$\alpha_{22} = \frac{l_1^2 (l - l_1)^2}{3EI_1 l} \quad (23)$$

$$\alpha_{33} = \frac{l_3^2 (l_1 + l_2)^2}{3EI_2 l} \quad (24)$$

The Dunkerley method disassembles the shaft in several sections. The squared reciprocal critical speed is calculated for every single section and summed up. The small overlaying end of the shaft is neglected in this approximation calculation in order to simplify it. In addition it is assumed that it does not affect the frequency, due to its very low weight. The following picture shows the allocation of the sections. Every section is denoted by its belonging mass. The mass m_1 was intentionally set to the end of the shaft, due to model building purposes. (Holzweißig & Dresig, 1982)

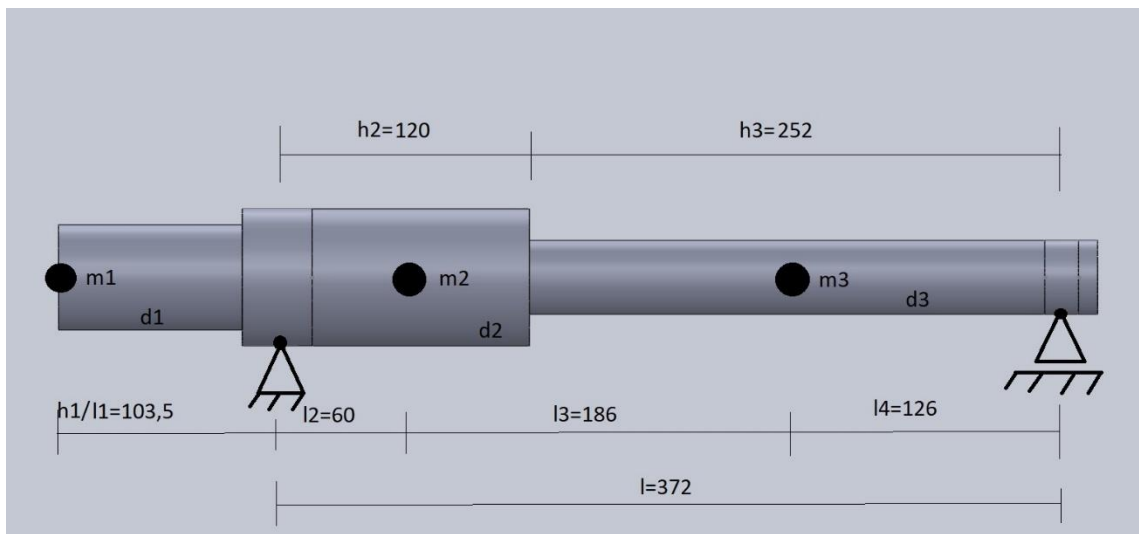


Figure 19 Sketch for control calculation of blank shaft

The mass is calculated by (Kolev & Zimmermann, 2009):

$$m_i = \rho * V_i \quad (25)$$

$$m_1 = 7850 \frac{kg}{m^3} * \frac{\pi}{4} * (0,05m)^2 * 0,1035m$$

$$m_1 = \underline{1,6 \text{ kg}}$$

$$m_2 = 7850 \frac{kg}{m^3} * \frac{\pi}{4} * (0,065m)^2 * 0,12m$$

$$m_2 = \underline{\underline{3,13 \text{ kg}}}$$

$$m_3 = 7850 \frac{\text{kg}}{\text{m}^3} * \frac{\pi}{4} * (0,035\text{m})^2 * 0,252\text{m}$$

$$m_3 = \underline{\underline{1,9 \text{ kg}}}$$

The area moment of inertia is calculated by (Kolev & Zimmermann, 2009):

$$I_i = \frac{\pi}{64} * d_i^4 \quad (26)$$

$$I_1 = \underline{\underline{30,68*10^4 \text{ mm}^4}}$$

$$I_2 = \underline{\underline{87,62*10^4 \text{ mm}^4}}$$

$$I_3 = \underline{\underline{73,66*10^3 \text{ mm}^4}}$$

$$I_a = \underline{\underline{19,59*10^4 \text{ mm}^4}}$$

$\frac{1}{\omega^2_{Kl}}$ is now calculated by (18) with the average diameter $d_a = 44,7\text{mm}$, which is built of the two diameters between the bearings and their belonging lengths.

$$\frac{1}{\omega^2_{Kl}} = \frac{(0,372\text{m})^4}{(0,0447\text{m})^2} * \frac{2*7850 \frac{\text{kg}}{\text{m}^3}}{\pi*2,1*10^{11} \frac{\text{kgm}}{\text{s}^2\text{m}^2}} = \underline{\underline{2,281*10^{-7} \text{ s}^2}}$$

$\frac{1}{\omega^2_{C1}}$, $\frac{1}{\omega^2_{C2}}$ and $\frac{1}{\omega^2_{C3}}$ is calculated by (17) and by means of (22), (23) and (24)

$$\frac{1}{\omega^2_{C1}} = \left(\frac{372\text{mm}*(103,5\text{mm})^2}{3*2,1*10^8 \frac{\text{kgmm}}{\text{s}^2\text{mm}^2} * 19,59*10^4 \text{ mm}^4} + \frac{(103,5\text{mm})^3}{3*2,1*10^8 \frac{\text{kgmm}}{\text{s}^2\text{mm}^2} * 30,68*10^4 \text{ mm}^4} \right) * 1,6\text{kg}$$

$$\frac{1}{\omega^2_{C1}} = \underline{\underline{6,084*10^{-8} \text{ s}^2}}$$

$$\frac{1}{\omega^2_{C2}} = \frac{(60\text{mm})^2*(372\text{mm}-60\text{mm})^2}{3*2,1*10^8 \frac{\text{kgmm}}{\text{s}^2\text{mm}^2} * 87,62*10^4 \text{ mm}^4 * 372\text{mm}} * 3,13\text{kg}$$

$$\frac{1}{\omega^2_{C2}} = \underline{\underline{5,342*10^{-9} \text{ s}^2}}$$

$$\frac{1}{\omega^2_{C3}} = \frac{(126\text{mm})^2*(60\text{mm}+186\text{mm})^2}{3*2,1*10^8 \frac{\text{kgmm}}{\text{s}^2\text{mm}^2} * 73,66*10^3 \text{ mm}^4 * 372\text{mm}} * 1,9\text{kg}$$

$$\frac{1}{\omega^2 c_3} = \underline{\underline{1,057 \cdot 10^{-7} \text{ s}^2}}$$

The equation (16) transformed to ω_c delivers finally the desired critical speed of

$$\omega_c = \underline{\underline{1581,1 \text{ Hz}}}$$

The approach according to Dunkerley gives the lower limit of the first eigenfrequency and is about 4% below the exact solution. If 4% are added to the 1581,1 Hz, the exact solution should be around 1644,4 Hz. This value comes very close to the 1635,8 Hz of the simulation. Hence the result of the simulation seems reasonable. Nevertheless, this approach was found empirically and does not deliver an exact solution, neither if 4 % are added to the calculated value. (Haberhauer & Bodenstein, 2009)

In order to check the sufficiency of the element size, another simulation is carried out with an adjusted element size. This simulation is run with the half element size (6mm) of the previous simulation. The number of elements is now 11294. The smaller elements deliver a frequency of 1635 Hz for the second mode, what is only a difference of 0,8 Hz. The change in the frequency is negligible small. This shows that the element size is reasonable.

Even though the mesh refinement delivered frequencies that were very close to each other, another element type is checked for the simulation. This time, the element type tetrahedron is used for the simulation. Since tetrahedrons are not so accurate, more elements are needed to get a sufficiently accurate result. Smaller elements with a size of 8 mm are chosen from the beginning. The entire model is now filled with 2173 tetrahedrons, which deliver a frequency of 1645,2 Hz. This is 10,2 Hz more than the Hex elements delivered. It is assumed that the number of the elements is not sufficient yet. A second simulation with an element size of 4 mm is carried out. The model consists now of 9071 elements and delivers the result of 1638,5 Hz. This is still about 3,5 Hz far from the simulation result with the Hex elements, therefore another simulation is carried out with an element size of 2 mm. The entire model is now filled with a total number of 37758 elements, what leads to a result of 1635,3 Hz. Now it can be seen that the result

converges against a certain value. This shows that the result cannot be varied very much. Hence the result of 1635 Hz is taken as the final result.

3.2.2 Simulation of the shaft in the first gear

In this simulation not only the shaft but also the rotor of the electric motor, the input shaft and the planetary gear is included. The model for this simulation was prepared in 3.1.2.

In the simulation software a so-called point mass is now inserted to the model. This point mass has the feature that mass moments of inertia can be allocated to the revolution around each axis of the coordinate system. The point mass is now exactly on the centre axis of the shaft. The distance between the point mass and the loose bearing is 50 mm. The behaviour is set to rigid. This means that the surface which the point mass is attached to, does not deform. It mirrors the behaviour of the real shaft pretty well, since the shaft is massive and the tothing stiffens the surface additionally in that area. In the figure below the introducing area of the mass moment of inertia is highlighted in red. It represents the area where the counter part of the jaw clutch engages in the tothing of the shaft.

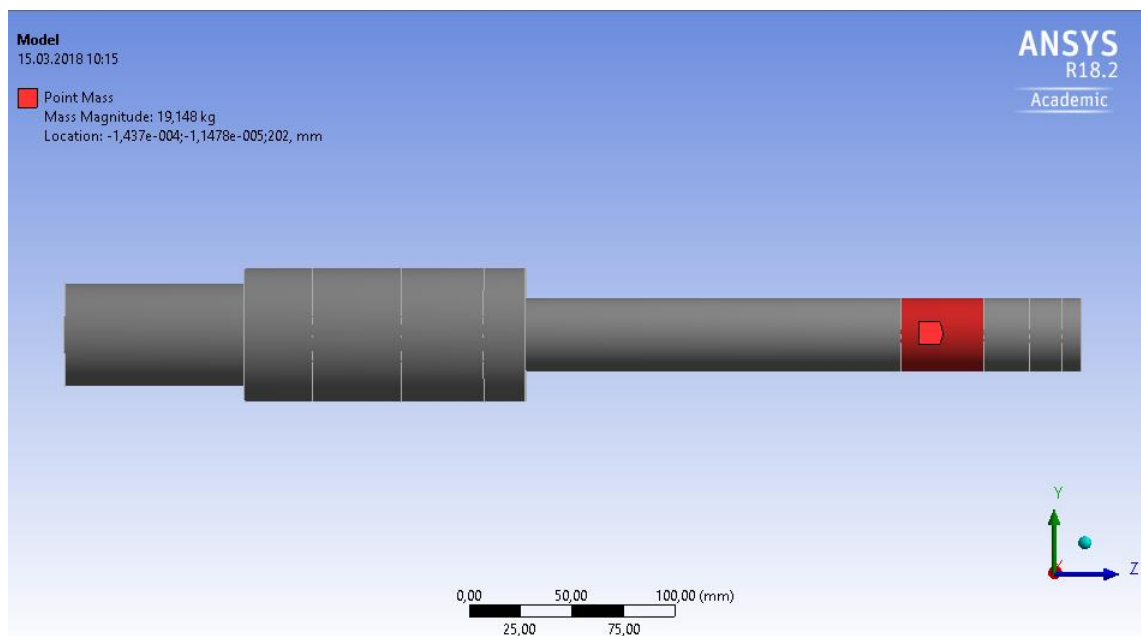


Figure 20 **Model with point mass**

All settings from the previous simulation like material, boundary conditions or the mesh remain the same. The only setting that is new is the previous described point mass.

The first mode that is shown is the rigid body mode again with a frequency of 0 Hz. The second mode shows a bending vibration with a frequency of 712 Hz. In the mode shape animation the shaft elongates up and down, as expected. The third mode elongates perpendicular to the second one at a frequency of 713,5 Hz. In the fourth mode a torsional vibration of the shaft is visible. The shaft vibrates now at 802,2 Hz.

The behaviour of the material in the bearings is not checked again because the boundary conditions were not changed.

Another approximation calculation according to Dunkerley is carried out to have an orientation value. For that, the results of every section in chapter 3.2.1 are kept and the formula (16) is extended by (Holzweißig & Dresig, 1982):

$$\frac{1}{\omega_i^2} = J_i * \beta_{ii} \quad (27)$$

with the influence coefficient β_{33} and the reduced mass moment of inertia J_{red} for the position that can be seen in Figure 1. The terms with the stiffness of the bearing is already neglected, since the bearings are considered infinitely stiff. (Holzweißig & Dresig, 1982)

$$\beta_{33} = \frac{l_6^3 + l_5^3}{3EI_3h_3^2} \quad (28)$$

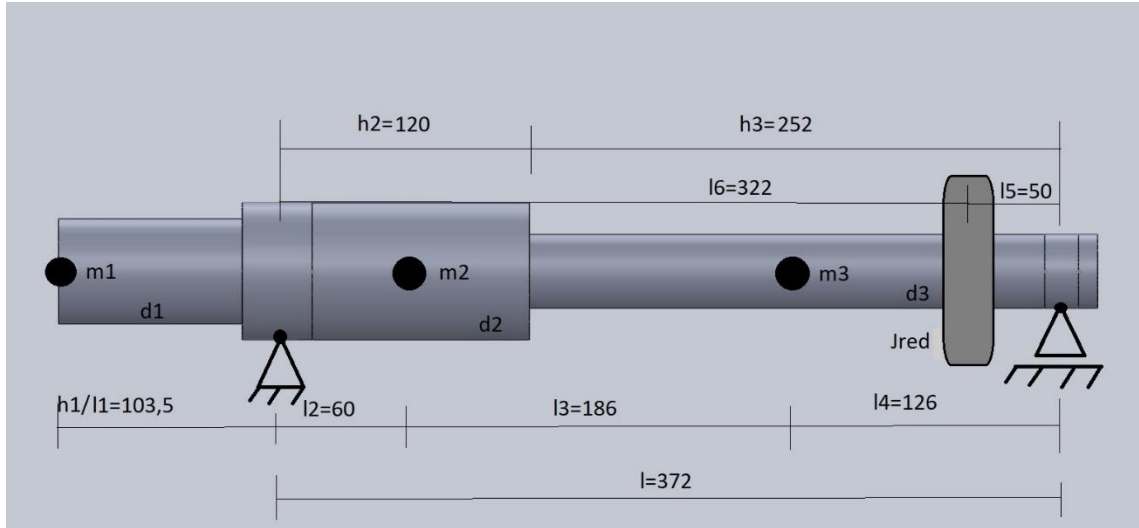


Figure 21 Sketch for control calculation of shaft in the first gear

$$\frac{1}{\omega^2_4} = J_{red} * \beta_{33}$$

$$\frac{1}{\omega^2_{C4}} = 141457,7 \text{ kgmm}^2 * \frac{(322\text{mm})^3 + (50\text{mm})^3}{3*2,1*10^8 \frac{\text{kgmm}}{\text{s}^2\text{mm}^2} * 73,66*10^3 \text{mm}^4 * (252\text{mm})^2}$$

$$\frac{1}{\omega^2_{C4}} = \underline{\underline{1,609*10^{-6} \text{ s}^2}}$$

Now the result of the term $\frac{1}{\omega^2_{C4}}$ is added to the results of the terms in Formula (16) and transformed to ω_c , what delivers:

$$\omega_c = \underline{\underline{705,6 \text{ Hz}}}$$

According to the control calculation, the value of 705,6 Hz is the lower limit for the first eigenfrequency. If 4% are added the exact value should be in a range of about 733,8 Hz. The first eigenfrequency of the simulation has the value of 712 Hz, which is not that close to the 733,8 Hz. But it is still above the lower limit of 705,6 Hz. Two other simulations are carried out in which the solver was not controlled by the program. For one simulation the direct solver is taken and for the other one the iterative solver. Both solvers delivered the exact same result. Thus the simulated value and the simulation is assumed to be correct.

A mesh refinement is not carried out again, since the mesh is still the same and a sufficiency of the mesh was already tested in 3.2.1

3.2.3 Simulation of the shaft in the second gear

All settings from the previous simulation like material, boundary conditions or the mesh remain the same, just the position of the point mass is changed. The new position of the point mass is now at a distance of 289 mm from the middle of the loose bearing. The properties of the point mass remain the same.

After solving the problem, the simulation shows the rigid body mode with 0 Hz. The second mode is a bending vibration. The mode shape looks similar to the one of the previous simulation. However, in this simulation the mode takes place at a value of 1140,4 Hz. In the next mode the shaft elongates perpendicular to the second mode with a frequency of 1140,7 Hz. The fourth mode has the eigen-frequency of 2037,3 Hz and is also a bending vibration.

For the control calculation the approach according to Dunkerley is used again. The formula (27) is used by means of (28) with the lengths l_7 and l_8 to calculate the natural frequency of the sub-system of the reduced mass moment of inertia.

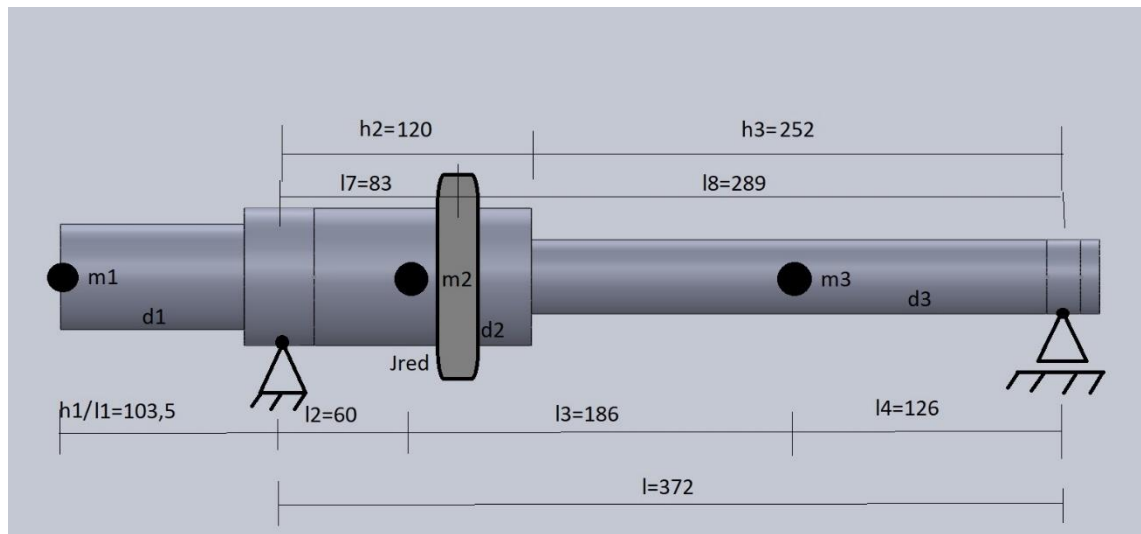


Figure 22 Sketch for control calculation of shaft in second gear

$$\frac{1}{\omega^2_{C5}} = 141457,7 \text{ kgmm}^2 * \frac{(83\text{mm})^3 + (289\text{mm})^3}{3 * 2,1 * 10^8 \frac{\text{kgmm}}{\text{s}^2 \text{mm}^2} * 87,62 * 10^4 \text{mm}^4 * (120\text{mm})^2}$$

$$\frac{1}{\omega^2_{C5}} = \underline{\underline{4,397 * 10^{-7} \text{ s}^2}}$$

Now the result of the term $\frac{1}{\omega^2_{C5}}$ is added to the results of the terms in Formula (16) and transformed to ω_c , what delivers:

$$\omega_c = \underline{\underline{1091,3 \text{ Hz}}}$$

4% are added to this value, so the exact value should be in a range of about 1134,9 Hz. This is not so far from the result of the simulation with 1140,4 Hz. Hence, the simulation seems reasonable.

3.3 Simulation in SolidWorks

For the simulation with the SolidWorks tool, the models were created in SolidWorks, according to 3.1.1 edited and directly used in the simulation tool.

3.3.1 Simulation of the shaft

The preparation of the models is omitted since the models for this analysis are the same as in the chapter 3.2 simulation with ANSYS. The preparation is carried out in chapter 3.1.

As material, the actual steel 42CrMo4 was selected. All mechanical properties are the same as in the other simulation, besides the density. The density is only 7800 kg/m³, unlike in ANSYS 7850 kg/m³. This should increase the frequency a little.

The element type is not selectable in the SolidWorks simulation tool. There are only tetrahedron elements available. The settings for these elements were left in standard settings.

In order to get simulation results, that are comparable to some degree, a mesh with approximately the same amount of elements is created. The mesh consists now of tetrahedron elements with an element size of 8,7 mm. It is not possible to achieve the same amount of elements with the same element size as in 3.2.1. This is due to the different settings for the mesh generation like Relevance Centre and Smoothing. The entire model is now filled with 8888 elements.

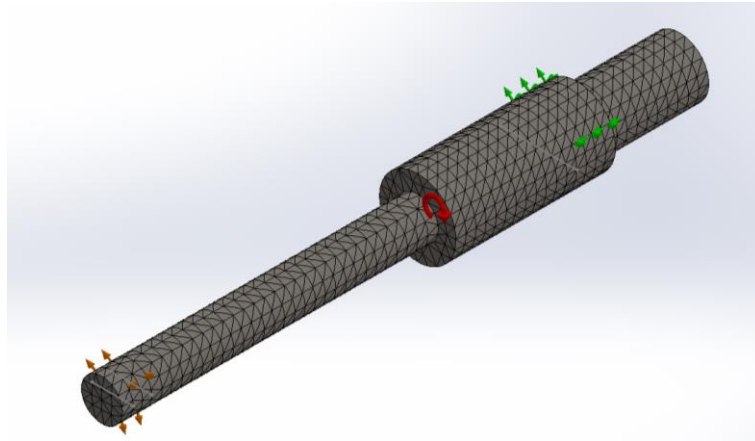


Figure 23 **Meshed model with tetrahedrons**

In this simulation, there is also a very small rotational velocity inserted. This is necessary to overcome the problem of numerical instability that the solver cannot deal with. In SolidWorks this feature is called Centrifugal Force. The Centrifugal Force that revolves around the Z- axis has now an angular velocity of 3 rad/s and an angular acceleration of 0.

The simulation was run and the list of the resonant frequencies shows for the first mode the frequency of 0,4 Hz. But the first amplitude that is shown in the mode shape animation is the second mode. It delivers a value of 1649,8 Hz. This result is not so far from the value 1644 Hz of the control calculation carried out in 3.2.1. The density in the control calculation was not adjusted. In the mode shape animation, the elongated shaft moves up and down. The second mode delivers a frequency of the exact same value and the mode shape is the same. The third mode shows the frequency of 1650,9 Hz. The elongation is perpendicular to the first two modes that are shown. In the fourth mode, there is again the torsional vibration at around 2975 Hz. The behaviour of the boundary conditions is the same as in 3.2.1. On balance, the simulation seems reasonable.

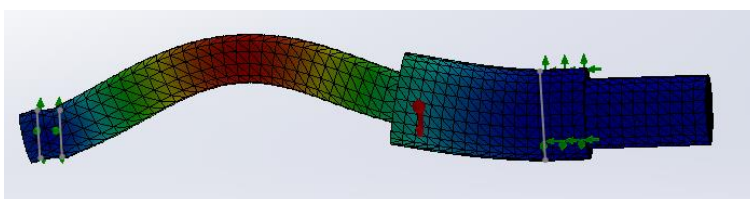


Figure 24 **Mode shape of first mode in SolidWorks**

In order to get a more accurate result the element size is adjusted. It was tried to obtain a similar amount of elements as in the last simulation with tetrahedron

elements in 3.2.1. The number of elements that were achieved by varying the size of the elements is 36698 with 5,2 mm. The simulation delivers then a frequency of 1643,7 Hz for the mode that is shown for amplitude 1. This value is taken as the final result. Other adjustments in the mesh settings were not done.

3.3.2 Simulation of the shaft in the first gear

All settings in this simulation were taken from the previous one. The only difference is the remote mass (point mass, highlighted pink in the figure) that is assigned to the exact same position and the same values as in 3.2.2.

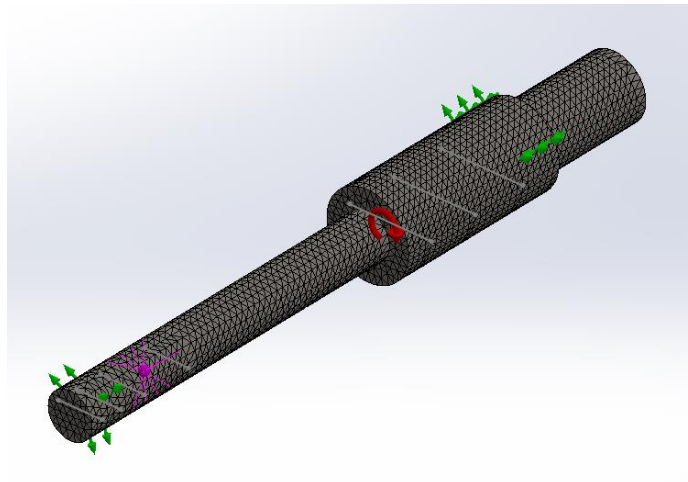


Figure 25 Point mass in SolidWorks

The list of resonant frequencies shows for the rigid body mode the frequency 0 Hz. This mode is also not shown in the simulation animation. The first amplitude that is shown in the animation is the second mode with 712,7 Hz. The elongation happens like in the previous simulation through an up and down motion. The first and second amplitudes that are shown are similar in elongation and frequency. The third amplitude with 714,3 Hz elongates perpendicular to the first and second one. In the fourth amplitude the elongation occurs torsionally at the end of the bigger diameter at a frequency of 812,9 Hz. The vibrational behaviour is similar to the behaviour in 3.2.2.

A mesh refinement is not carried out again, since the mesh is still the same and a sufficiency of the mesh was already tested.

3.3.3 Simulation of the shaft in the second gear

All settings are the same as in the previous simulation, only the position of the point mass (Remote Mass) is changed. It has the same position as the Mass Point in 3.2.3.

For the first mode, which is the rigid body mode, 0 Hz were calculated by the solver. This mode is skipped again in the animation of the mode shapes. The first and the second mode that are shown elongate in a similar way like in the previous simulation, but at a frequency of 1174 Hz. This value is quite far from the value of the control calculation. Thereupon, both solvers, the iterative and the direct solver were tried out. But both solvers delivered the exact same value. The third mode elongates perpendicular to the two previous ones with a frequency of 1174,9 Hz. The next mode occurs at 2109,8 Hz and has two elongations in opposite directions. The behaviour of the shaft is similar to the behaviour in the simulation that was carried out in 3.2.3.

4 Comparison of ANSYS and SolidWorks

4.1 Material library

The material library in ANSYS is called Engineering Data and offers a rather sparse range of materials compared to SolidWorks. Steel grades include only a few steels that can be used in the simulation. But it is possible to create a customized material by inserting the necessary values for the mechanical properties. Other materials like non-ferrous metals, plastics, ceramics or concrete are also available in a limited range.

The material library in SolidWorks is very well equipped with materials. SolidWorks offers an own library with the name SOLIDWORKS Materials that includes materials like steel, iron, different non-ferrous metal alloys but also materials like plastics, rubber or wood. Additionally there is a library with the name SOLIDWORKS DIN Materials, which includes a wide range of materials according to the German standard DIN.

4.2 Mesh and element type

In the ANSYS software there are more than 60 different element types for mechanical simulation available. Among them are beam elements, shell elements and continuum elements with a different number of nodes. Different kinds of elements can be used to mesh different sections of a structure. Additionally there are plenty of options provided to generate a more accurate mesh. This can be done for instance by classifying the Relevance Centre, the Transition from coarse to fine mesh, the Element Size and much more. Not only a wide range of possi-

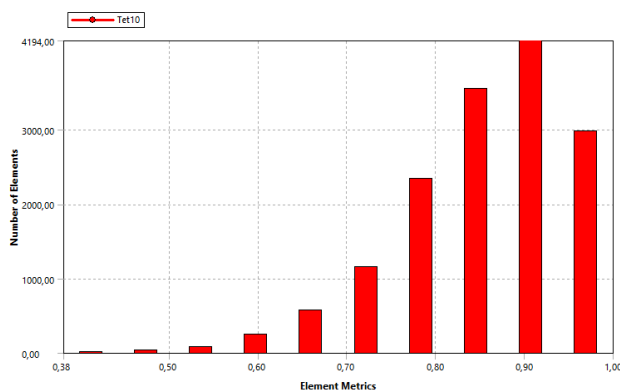


Figure 26 Mesh metric diagram

bilities to edit the mesh are available but also several functions to control the created mesh. In the option mesh metric, the general quality of the elements or the angles of the elements can be checked. The figure shows a diagram to the mesh quality. On the X- axis the quality factor that

reaches from 0 to 1 can be seen. The factor is the ratio of volume and edge length of the corresponding element. A value of 1 represents a perfect shaped cube or tetrahedron. (ANSYS Inc., 2017)

In SolidWorks, there are only tetrahedron elements of first and second order for solid structures available. While creating the mesh, it was determined that the amount of elements is not achievable by inserting the same elements size. This is due to the different settings of mesh parameters in both software. A few settings for the mesh parameters are available to improve the mesh and affect the result. For instance, Curvature-based mesh and Blended curvature-based mesh are settings to refine the mesh locally and can be helpful to overcome meshing failure. In addition, the number of Jacobian points for the elements are adjustable. Besides the tetrahedron elements, triangular shell elements of first and second order and beam and rod elements are available. (Brand, 2016; Dassault Systèmes SolidWorks Corp., 2018)

4.3 Boundary conditions

It can be selected between eleven different supports in the ANSYS software, amongst others Fixed Support, Displacement, Remote Displacement or Cylindrical Support. If a Cylindrical Support is required for the simulation the geometry has to be edited or completely generated in ANSYS SpaceClaim. This is an integrated tool similar to a 3D CAD-Software, which can also be used to remove small radii, edges and so on to clean the geometry. This editing is necessary to generate a cylindrical face. If the model is directly inserted as an IGES file or a file of the 3D-Software the cylindrical face is split in two halves. ANSYS delivers also the rigid body modes if the entire structure is unconstraint in a modal analysis. The subsequent mode delivers then the natural frequency of the structure without any boundary conditions. A point mass can be assigned to a surface and is automatically located in the middle of the volume or the face. The location can be changed by inserting values for the position according to the local coordinate system. The behaviour of the point mass can be chosen between rigid, coupled, deformable and beam. The mass and mass moment of inertia around each axis can be assigned.

The main supports in SolidWorks are constricted to three different kinds, Fixed Geometry, Roller/ Slider and Fixed Hinge. But also Elastic Supports and Contacts are possible to simulate. In the option advanced, it is possible to specify the constraints for instance by constraining a certain shaped face like a sphere or cylinder. In SolidWorks it is not possible to leave one degree of freedom unconstraint. That leads to numerical problems for the solver. In the case of the shaft, it was necessary to insert at least a small rotational velocity or a weak spring, because the solver cannot deal with rigid body modes. Inserting a point mass is carried out by selecting a surface and defining the location according to the local coordinate system of the part. The required values for the point mass are the mass and the mass moment of inertia of at least one axis. Mass moment of inertias can be assigned around each axis like in ANSYS. Unlike in ANSYS the behaviour of the point mass is limited to rigid. (Dassault Systéms SolidWorks Corp., 2018)

4.4 Solvers

Five different solvers are selectable in ANSYS. It can be chosen between Direct and Iterative solver and Unsymmetric, Supernode and Subspace. Additionally there are options to modify the solution process by changing settings in Solution Progression, Output Controls, Launch Controls and Solver Settings panel. (ANSYS Inc., 2017)

In SolidWorks, there are three solvers available, two direct solvers and one iterative solver. The Intel Direct Sparse solver works with improved speed for problems that are solved in-core. (Dassault Systéms SolidWorks Corp., 2018)

4.5 Results

Case	Result ANSYS in Hz	Result Solid-Works in Hz	Control calculation (+4%) in Hz
Blank shaft	1635,3	1643,7	1644,4
First gear	712	712,7	733,8
Second gear	1140,4	1174	1134,9

Table 2: comparison of the results

Table 2 shows the simulated values of the first mode and the values of the control calculation for every single simulation case. In the first case, both values of ANSYS and SolidWorks are close to the result of the control calculation. The value that was achieved with SolidWorks is, as expected, a bit higher than the ANSYS value. It matches almost exactly the control value.

In the second case, both simulation results are almost equal but relatively far away from the control value. But both values are above the lower limit that was calculated by means of the Dunkerley approach. The value of SolidWorks is again slightly higher than the ANSYS value.

In the last case, both values are above the value of the control calculation. The value of ANSYS is just 5,5 Hz higher but still in a reasonable range whereas the SolidWorks value is 39,1 Hz far from the calculation result. Hence the SolidWorks value is about 3,5% over the result the control calculation.

The simulation values are spread around the control values without any recognisable pattern. This shows the results are not really comparable regarding their accuracy. The deviations might be due to the different amount of elements used in each software. In 3.3.1 it was detected that the amount of elements is not so finely adjustable, what can lead to deviations in the simulation values.

5 Conclusion

The conclusion section states the limitations of this project, makes recommendations on how this topic could be investigated further, reflects on the simulations and closes with a final conclusion.

5.1 Limitations

One limitation of this project what is due to the limited timeframe is that only analyses of a single part were carried out. Therefore only one type of support was compared. Furthermore the analysis type was constricted to a vibration analysis. Both software offer a wide range of possibilities of analysis types.

Another limitation is that the achieved results of both software are not really comparable with regard to their accuracy. The structure that was used for the simulation is already rather complex, what can already lead to unmeant differences in the frequency value. In addition, it makes the control calculation more difficult as well.

5.2 Recommendations

For further investigations in this field it is recommended to simulate several structures, which have different boundary conditions in order to use more features of the software. Additionally, other analysis types than a modal analysis for instance a static analysis or a buckling analysis can be carried out. With this measure the field of investigation can be spread more widely.

For a comparison of the results regarding to their accuracy the simulation of simple shaped structures is recommendable. This ensures easy and clearly arranged control calculations with precise and meaningful results, which can be used as absolute values to compare the simulation results.

5.3 Reflections

This project has been very educational and in some parts knowledge could be consolidated. Despite of some previous knowledge in FEM, the theory review part was illuminating and generated a deeper understanding of the topic. Even

though, a lot of preparation exercises had been done before the thesis was started, it was hard to select the right settings in ANSYS and to handle the program. Nevertheless, the simulation results were always within a reasonable range of the control calculation. The comparison has pointed out that the major differences in both software can be found in the options of the mesh generation and the options for the boundary conditions.

5.4 Final conclusion

Ultimately, the project has shown that both software can deliver adequate results, but the ways to achieve them are different. One way offers more opportunities than the other one. This can be useful to reproduce special cases of reality or to generate a tailored mesh for a complex structure. However, for the structure that was analysed in this thesis both software are totally sufficient.

The simulation tool of SolidWorks is a good opportunity for designers to make a quick and simple analysis after designing or in between, especially since the tool is integrated in the CAD-Software. Moreover, it is excellent for beginners in FEM to learn the basics, due to its simplicity compared to a professional FEM-Software. The learning user is not confronted with a huge range of setting options, which vary the simulation result.

In return, ANSYS offers a wealth of options that have influence to the simulation result. The use of a professional FEM-Software requires a deep knowledge in FEM-Simulation. Additionally the user should get along quite well with the software and should know which settings have which impacts on the results. For this reason it requires highly skilled engineers who know exactly what they do. It is also possible to change the geometry of the structure in the integrated CAD-Tool before and after a simulation.

This thesis work has resulted in a much better understanding of Finite Elements Analysis, vibrations and machine dynamics. Additionally, the skills to operate ANSYS Workbench and the simulation tool of SolidWorks have been improved quite a bit. The thesis was quite a challenge and not only knowledge and skills that are directly related to the thesis topic have been improved, but also the ability to deal with problems and helping oneself.

Figures

Figure 1	Local coordinate system (Dominico, 2017)	10
Figure 2	Global coordinate system (Dominico, 2017).....	10
Figure 3	Assembled structure (Dominico, 2017)	10
Figure 4	Element types (Rusu, 2013).....	11
Figure 5	Mass distribution tetrahedron elements (Rieg, et al., p.150- 151, 2014)	12
Figure 6	1 st mode of a plate construction	20
Figure 7	2 nd mode of a plate construction.....	20
Figure 8	2 nd mode of a connecting rod, top view	20
Figure 9	1 st mode of a connecting rod, front view.....	20
Figure 10	Buckling behaviour of structures (Pataik & Hopkins, 2004).....	20
Figure 11	State of stability (Pataik & Hopkins, 2004)	21
Figure 12	Section view of electric motor.....	23
Figure 13	Cleaned 3D model for simulation	24
Figure 14	Uncleaned 3D model of the shaft	24
Figure 15	Model for reduction of mass moment of inertia	25
Figure 16	Meshed shaft.....	27
Figure 17	Supports of the shaft	28
Figure 18	Mode shape of second mode	29
Figure 19	Sketch for control clacualtion of blank shaft	30
Figure 20	Model with point mass.....	33
Figure 21	Sketch for control calculation of shaft in the first gear	35
Figure 22	Sketch for control calculation of shaft in second gear	36
Figure 23	Meshed model with tetrahedrons	38
Figure 24	Mode shape of first mode in SolidWorks	38
Figure 25	Point mass in SolidWorks.....	39
Figure 26	Mesh metric diagram.....	41

References

Anon., 2011. *Stochastic Simulation and Lagrangian Dynamics*. [Online] Available at: <http://stochasticandlagrangian.blogspot.fi/2011/07/what-does-shape-function-mean-in-finite.html>

[Accessed 30. January 2018].

ANSYS Inc., 2017. *ANSYS Help Guid*, s.l.: s.n.

Brand, M., 2016. *FEM-Praxis mit SolidWorks*. 3rd ed. St. Gallen, Schweiz: Springer Vieweg.

CADFEM GmbH, 2015. *www.cae-wiki.info*. [Online] Available at: http://www.cae-wiki.info/wikiplus/index.php/Konzentrierte_Massenmatrix

[Accessed 27. February 2018].

CAEFEM GmbH, 2015. *www.cae-wiki.info*. [Online] Available at: http://www.cae-wiki.info/wikiplus/index.php/Konsistente_Massenmatrix

[Accessed 27. February 2018].

Dassault Systéms SolidWorks Corp., 2018. *SolidWorks Help- Analysis Solvers*, s.l.: s.n.

Dassault Systéms SolidWorks Corp., 2018. *SolidWorks Help- Mesh Property Manager*, s.l.: s.n.

Dassault Systéms SolidWorks Corp., 2018. *SolidWorks Help- Peventing rigid Body motion.*, s.l.: s.n.

Dölle, V. & Heiny, D., n. d.. *SimScale*. [Online] Available at: <https://www.simscale.com/blog/2016/08/how-to-choose-solvers-for-fem/>

[Accessed 29. January 2018].

Dominico, P. D. S., 2017. *Script Practical Introduction to FEM*. s.l.:s.n.

Haberhauer, H. & Bodenstein, F., 2009. Elemente der drehenden Bewegung. In: *Maschinenelemente Gestaltung, Berechnung, Anwendung*. s.l.:Springer Verlag.

Heid Antriebstechnik Produktion & Handel GmbH, n. d.. www.heid-antriebstechnik.at. [Online]

Available at: http://www.heid-antriebstechnik.at/pdf/technische_erlauterungen.pdf

[Accessed 26. February 2018].

Holzweißig, F. & Dresig, H., 1982. *Lehrbuch der Maschinendynamik Maschinendynamische Probleme und ihre praktische Lösung*. 2. ed. Altenburg: Springer Verlag.

Hutton, D. V., p. 402- 407, 2004. *Fundamentals of Finite Elements Analysis*. 1. ed. New York: The McGraw-Hill companies Inc..

Klein, B., 2014. *Grundlagen und Anwendungen der Finite-Element-Methode im Maschinen- und Fahrzeugbau*. Calden: Springer Verlag.

Kolev, P. D.-I. E. & Zimmermann, P. D.-I. K., 2009. *Formelsammlung Technische Mechanik*. Schmalkalden: s.n.

Liu, G. R. & Queck, S. S., 2003. *The Finite Element Method: A Practical Course*. s.l.:Butterworth-Heinemann.

Pataik, S. N. & Hopkins, D. A., 2004. *Strength of Materials: A Unified Theory*. s.l.:Butterworth-Heinemann.

Petyt, M., 2010. *Introduction to Finite Element Vibration Analysis*. 2 ed. New York: Camebridge Uiversity Press.

Rieg, F., Hackenschmidt, R. & Alber-Laukant, B., 2014. *Finite Element Analysis for Engineers*. Munich: Carl Hanser Verlag.

Rieg, F., Hackenschmidt, R. & Alber-Laukant, B., p.150- 151, 2014. *Finite Element Analysis for Engineers*. Munich: Carl Hanser Verlag.

Rusu, C., 2013. *FEA for all*. [Online]
Available at: <http://feaforall.com/why-do-fea-engineers-use-1d-2d-3d-elements/>
[Accessed 30. January 2018].

Steinke, P., 2015. *Finite-Elemente-Methode Rechnergestützte Einführung*. 5. ed.
Steinfurt: Springer Verlag.

Wallin, M., 2014. *A Finite Element tool for buckling analysis*. Linköping: s.n.

Weidner, P. D.-I. G., 2016. *Vorlesungsskript Getriebetechnik*. Schmalkalden: s.n.

Wikimedia Foundation Inc., 2007. *Wikipedia*. [Online]
Available at:
https://de.wikipedia.org/wiki/Gau%C3%9Fches_Eliminationsverfahren#Eigenschaften_des_Verfahrens
[Accessed 28. January 2018].

Wikimedia Foundation Inc., 2018. *Wikipedia*. [Online]
Available at: https://de.wikipedia.org/wiki/Finite-Elemente-Methode#Verarbeitung:_Gleichungsl%C3%B6ser
[Accessed 28. January 2018].

Zehn, P. D.-I. h. M., 2009. *Vorlesungsskript Mechanik 2*. Berlin: s.n.

Appendices

Appendix 1 Drawing of the jaw clutch

Appendix 2 Drawing of the output shaft

